NASA Technical Memorandum 85734

USER'S MANUAL FOR GAMNAS--GEOMETRIC AND MATERIAL NONLINEAR ANALYSIS OF STRUCTURES

John D. Whitcomb and B. Dattaguru

JANUARY 1984

(NASA-TM-85734) USER'S MANUAL FOR GAMNAS: GEOMETRIC AND MATERIAL NONLINEAR ANALYSIS OF STRUCTURES (NASA) 40 p HC A03/MF A01 N84-15200
CSCL 11D Unclas
G3/24 18041

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

Introduction

GAMNAS (Geometric and Material Nonlinear Analysis of Structures) is a two-dimensional finite element stress analysis program. The program was developed to support fracture mechanics studies of debonding and delamination (refs. 1-3). Options include linear, geometric nonlinear, material nonlinear, and combined geometric and material nonlinear analysis.

The purpose of this manual is to document the theoretical basis of GAMNAS and to provide instruction in the use of the program. Details of the program organization and logic are presented in order to guide the user who needs to modify the code to meet some special need. Familiarity with linear finite element analysis is assumed.

First, theoretical aspects of GAMNAS are presented. Then program specifications, such as allowable problem size, are given. Next the program organization is described. Finally, the required input data is described. Brief descriptions of the subroutines and major program variables are given in Appendix A. Appendix B gives input data and results for several sample problems. Appendix C briefly discusses error messages and possible debug strategies.

Successful use of any finite element program depends largely on the ability of the analyst to qualitatively predict the response of a configuration before attempting detailed finite element analysis. This insight will generally be based on experience and possibly some strength of materials arguments. Also, very coarse finite element models can be useful. Insight is particularly important for nonlinear analysis, in which questions of convergence, uniqueness of solution, and solution strategy must be addressed. Hence, the user should become thoroughly familiar with the theoretical basis of GAMNAS and then gain experience by analyzing a variety of simple configurations before attempting to analyze a complex configuration.

Nomenclature

$[\bar{B}]$	incremental strain-displacement matrix
$[D^*]$	elasto-plastic constitutive matrix
E	Young's modulus
F	yield surface function
G	total strain-energy-release rate
G_I, G_{II}	mode I and mode II components of strain-energy-release rate
I	moment of inertia
$[\bar{K}]$	transformed global stiffness matrix
$[K_o],[K_T]$	linear and tangential stiffness matrices, respectively
M	moment
$P_{\bar{x}}, P_{\bar{y}}$	forces transmitted through crack tip in the $\bar{x}$ and $\bar{y}$ directions
$\{R\}$	applied load vector
$\{\bar{R}\}$	transformed applied load vector
$[T]$	transformation matrix
u,v	displacements in x and y directions, respectively
$\bar{u},\bar{v}$	displacements in $\bar{x}$ and $\bar{y}$ directions, respectively
V	volume
x,y	rectangular Cartesian coordinates
$\bar{x},\bar{y}$	rotated rectangular Cartesian coordinates
β	fraction of strain increment required to reach yield surface
Δa	virtual crack closure length
$\{\Delta\delta\}$	increment to nodal displacement vector
$\{\Delta\varepsilon\}$	strain increment
$\{\Delta\bar{\varepsilon}\}$	strain increment required to reach yield surface
$\{\Delta\hat{\varepsilon}\}$	strain increment after reaching yield surface
$\{\delta\}$	nodal displacement vector
$\{\bar{\delta}\}$	transformed nodal displacement vector

$\varepsilon_x, \varepsilon_y, \varepsilon_{xy}$ normal strains in x and y directions and shear strain in xy plane, respectively

$\{\varepsilon\}$ strain vector

$\{\varepsilon\}_p$ plastic strain increment

λ proportionality constant

σ_{ef} effective stress, equal to $\left(\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x\sigma_y - \sigma_y\sigma_z - \sigma_x\sigma_z + 3\sigma_{xy}^2\right)^{1/2}$

σ_{ys} uniaxial yield stress

$\{\sigma\}$ stress vector

$\{\sigma_o\}$ stress vector before strain increment

$\{\sigma_1\}$ stress vector after strain increment

$\{\psi\}$ residual force vector

Theory

Governing Equations

This section outlines the theoretical basis for the GAMNAS computer code. First, geometric and material nonlinearity are discussed in general. Then application of the displacement based finite element method to nonlinear problems are discussed. The description given here follows closely that given in refs. 4 and 5, where details may be found.

Herein, geometric nonlinear analysis refers to an analysis which calculates strains using the Lagrangian nonlinear strain-displacement relations, eqns. (1)

$$\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2\right]$$

$$\varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right] \qquad (1)$$

$$\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial u}{\partial x}\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\frac{\partial v}{\partial y}$$

The second-order terms in eqns. (1) account for finite rotations. However, the strains are still assumed to be infinitesimal.

For material nonlinear analysis, the nonlinear relationship between stress and strain is defined incrementally, eqn. (2)

$$d\{\sigma\} = [D^*]\ d\{\varepsilon\} \tag{2}$$

The nonlinear elasto-plastic constitutive matrix $[D^*]$ is a function of the assumed yield surface and flow rule and the current stress state. GAMNAS uses the von Mises yield surface, eqn. (3) and a flow rule based on the normality principle, eqn. (4).

$$F = \left(\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x\sigma_y - \sigma_y\sigma_z - \sigma_x\sigma_z + 3\sigma_{xy}^2\right)^{1/2} - \sigma_{ys} \tag{3}$$

$$d\{\varepsilon\}_p = \lambda \frac{\partial F}{\partial\{\sigma\}} \tag{4}$$

The instantaneous uniaxial yield stress, σ_{ys}, in eqn. (3) is a function of the strain history and the specified uniaxial stress-strain curve. Three types of uniaxial stress-strain curves can be specified: elasto-plastic, bilinear, and Ramberg-Osgood. These are shown schematically in fig. 1. GAMNAS can only analyze plastic deformation of isotropic materials.

Application of the finite element method to nonlinear problems is very similar to that for linear problems. In both cases a system of equations is derived which expresses the equilibrium of internally generated forces in a body with externally applied forces, eqn. (5)

$$\{\psi\} = \int_{VOL} [\bar{B}]^T \{\sigma\}\ dV - \{R\} = 0 \tag{5}$$

In eqn. (5) $\{\psi\}$, $\{\sigma\}$, and $\{R\}$ are the residual force, stress, and applied load vectors, respectively. The integral is the vector of internally generated forces. The matrix $[\bar{B}]$ is the incremental strain-displacement matrix, as defined by eqn. (6)

$$d\{\varepsilon\} = [\bar{B}]\ d\{\delta\} \tag{6}$$

where $[\delta]$ is the nodal displacement vector, i.e., a list of u and v displacements at the nodes. For linear problems eqn. (5) is a linear set of equations with unknowns $\{\delta\}$.

For geometrically nonlinear problems eqns. (1) are used with eqn. (6) to derive $[\bar{B}]$. The matrix $[\bar{B}]$ is found to vary linearly with $\{\delta\}$, as is expected from the quadratic form of eqn. (1). The stresses $\{\sigma\}$ are linearly related to the strains, which vary quadratically with $\{\delta\}$. Hence, eqn. (5) is a set of cubic equations in $\{\delta\}$.

For elasto-plastic problems, the matrix $[\bar{B}]$ is independent of $\{\delta\}$, but the relationship between $\{\delta\}$ and $\{\sigma\}$ is a complicated nonlinear function. Furthermore, the relationship between $\{\delta\}$ and $\{\sigma\}$ is path (i.e., history) dependent. Hence, the solution of eqn. (1) for a desired load level is obtained by dividing the total load into a series of small load increments. For each load increment, the relationship between stress and strain is determined from eqn. (2).

For combined geometric and material nonlinearity, the nonlinear relationships for each are simply used together.

Iterative Solution

The governing equations, eqn. (5), are solved iteratively using modified Newton-Raphson methods (ref. 4). The basic Newton-Raphson method for the first load step is outlined below.

1. Obtain a linear solution using the linear stiffness matrix K_o:

$$\{\delta_o\} = [K_o]^{-1}\ \{R\}$$

2. Calculate residuals $\{\psi\}$ with eqn. (5)

3. Check for convergence. Stop if $\{\psi\}$ is sufficiently small.

4. Calculate tangential stiffness matrix, $[K_T]$

   (The tangential stiffness matrix is defined by the equation

   $[K_T]\ \{\Delta\delta\} = \{\Delta\psi\}$.)

5. Solve for correction to displacements

$$\{\Delta\delta\} = -\ [K_T^{-1}]\ \{\psi\}$$

6. Update displacements: $\delta = \delta + \Delta\delta$

7. Go to step 2.

If multiple load steps are used, only step 1 changes. After obtaining a converged solution for load step "i", the linear solution (i.e., the new step (1)) for the next load step is

$$\{\delta\}_{i+1} = \{\delta\}_i + [K_T]^{-1}\ \{\Delta R\}_{i+1} \tag{7}$$

where $\{\Delta R\}_{i+1}$ is the load increment.

Different versions of the Newton-Raphson technique described above were used for geometric nonlinear, material nonlinear, and combined nonlinear analysis in GAMNAS. The main differences are in the way the tangential stiffness matrix, $[K_T]$, is approximated. For geometric nonlinear analysis $[K_T]$ is updated every "NCYCLE" iterations, where NCYCLE is an input parameter. For material nonlinear analysis $[K_T]$ is approximated by the linear stiffness matrix $[K_o]$ for all iterations. For combined geometric and material nonlinear analysis $[K_T]$ is updated every "NCYCLE" iterations, but the linear stress-strain relations are used in calculating $[K_T]$. For combined nonlinear analysis the solution for each load increment begins with obtaining a converged solution in which no additional yielding is allowed. After obtaining this "transition" solution, iterations begin in which both geometric and material nonlinear effects are included. This procedure reduces spurious material yielding which can be an artifact of iterative solution procedures. This procedure will be discussed further in the discussion of the flowchart for the subroutine ITERATE.

Strain Energy Release Rates

GAMNAS has the option to calculate Mode I and Mode II strain energy release rates. Strain energy release rates are calculated using a virtual crack extension technique similar to that reported in ref. 6. This technique uses the forces transmitted across the crack tip and the relative displacements just ahead of the crack tip to determine the energy release rate. For geometrically nonlinear problems the forces and displacements are transformed to the local rotated coordinate system, as shown in fig. 2. Figure 2 also shows the equations used to calculate G_I and G_{II}. The strain energy release rate calculation is valid for linear and geometrically nonlinear analysis only. The program assumes the mesh around the crack tip is rectangular and that the crack is initially parallel to the x-axis. Near the crack tip the mesh must be symmetrical about the crack tip.

Boundary Conditions

The following boundary conditions can be prescribed in GAMNAS:

1. Nodal loads
2. Specified displacements
3. Equivalence of two or more displacements, e.g., $\delta_i = \delta_j$
4. Equivalence of one displacement and the negative of another displacement, e.g., $\delta_i = -\delta_j$

To prescribe a displacement $\delta_i = \delta_o$ the diagonal term of the i^{th} equation is replaced by a large number, 10^{30}, and the "load" term for the i^{th} equation is set to $10^{30}\ \delta_o$. To impose a multi-point constraint, i.e.,

$\delta_i = \delta_j$ or $\delta_i = -\delta_j$, the displacement and load vectors and the stiffness matrix are transformed (ref. 10). The transformation is best explained by example. Consider the linear system $[K]\ \{\delta\} = \{R\}$. Assume there are four nodal displacements. To impose the condition $\delta_1 = \delta_3$ a new displacement vector $\{\bar{\delta}\}$ is defined such that

$$\{\delta\} = [T]\ \{\bar{\delta}\}$$

$$\begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 1 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} \delta_1 - \delta_3 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix} \tag{8}$$

The new stiffness matrix $[\bar{K}]$ and load vector $\{\bar{R}\}$ are

$$[\bar{K}] = [T]^T\ [K]\ [T]$$

$$\{\bar{R}\} = [T]^T\ \{R\} \tag{9}$$

The new governing equations are $[\bar{K}]\ \{\bar{\delta}\} = \{\bar{R}\}$. Note that $\bar{\delta}_1 = \delta_1 - \delta_3$. Hence, to impose the condition $\delta_1 = \delta_3$, we need simply impose the condition $\bar{\delta}_1 = 0$.

When multi-point constraints are specified, the bandwidth generally increases. The increase in bandwidth depends on the node numbering scheme. Hence, the multi-point constraints should be considered when selecting the node numbering scheme.

Elements

GAMNAS uses the four-node isoparametric quadrilateral. This element is well known to perform poorly in modeling bending type deformation when exact integration is used. But the performance can be dramatically improved by using selective reduced integration. References 7 and 8 describe the procedure for linear problems. Reference 9 describes the procedure for geometrically nonlinear problems. The user can specify either full or selective reduced integration in the program.

Program Specifications

GAMNAS is written in Prime's extended version of FORTRAN 77. Core requirements are 604,000 16 bit words and compilation time is approximately 2 minutes on the Prime 750. Execution times will vary greatly depending on the particular finite-element model. The current maximum allowable values of the major parameters are given in the description of the input data. An in-core equation solver is used. Hence, the maximum problem size is limited by the memory of the computer being used.

Most of the core requirements are for holding the global stiffness matrix, "SN." The matrix SN is dimensioned (1300, 70), which permits 1300 degrees of freedom (650 nodes) and a bandwidth of 70. GAMNAS can be quickly modified using a text editor to change the maximum bandwidth and number of nodes. The required changes and the order the changes should be made are listed below:

1) Change the string "(1300,70" to "(XXX,YYY" everywhere, where XXX and YYY are the new number of rows and columns, respectively.
2) Change the string "(1300" to "(XXX" everywhere, where XXX is the new number of rows.
3) In subroutine INITIAL change the following two lines:

   MRANK = 1300 ← change 1300 to XXX

   MIBW = 70 ← change to 70 to YYY

where XXX and YYY are the new number of rows and columns in SN, respectively.

Program Organization

In this section the flow of GAMNAS is described. Flowcharts are given for the more complicated routines: the main program, ITERATE, and STRSCAL. Very brief description of the subroutines and the major program variables are given in Appendix A.

An annotated flowchart for the main program is shown in Figure 3. Only one proportional load vector is input. The different load numbers (LOADNUM) refer to the scale factor by which the load vector is multiplied. For each new load, a linear incremental solution is obtained in the main program before calling ITERATE to obtain the nonlinear incremental solutions. the linear solution for the first load step and all nonlinear solutions are output.

Figure 4 shows a flowchart for the subroutine ITERATE. The subroutine utilizes the modified Newton-Raphson technique described earlier to solve eqn. (5). Note that for combined geometric and material nonlinearity (i.e., ANALYS = CNONLIN), the routine GITER is called to obtain a transition nonlinear solution for the load increment, assuming no additional yielding occurs. Then ITERATE proceeds to determine the converged solution which includes both geometric and material nonlinearity. The tangential stiffness matrix is updated by calling STIFF. For just material nonlinearity (i.e., ANALYS = PNONLIN), STIFF is not called. For geometric or combined nonlinear analysis, STIFF is called every "NCYCLE" iterations.

Figure 5 shows a flowchart for the subroutine STRSCAL. STRSCAL calculates the incremental stress vector $\{\Delta\sigma\}$ corresponding to the calculated incremental strains $\{\Delta\varepsilon\}$. For linear material response, $\{\Delta\sigma\}$ is simply the product of the constitutive matrix [D] and $\{\Delta\varepsilon\}$. For nonlinear material behavior the relationship between $\{\Delta\varepsilon\}$ and $\{\Delta\sigma\}$ depends on the current stress state $\{\sigma_o\}$ relative to the yield surface and on the magnitude of the strain increment. The relative positions of the stress state and the yield surface is determined from eqn. (3). For convenience in the flowchart, the first term in eqn. (3) is defined to be the effective stress σ_{ef}. For an arbitrary stress state $\{\sigma\}$, the following relationships apply:

$\sigma_{ef}(\{\sigma\}) < \sigma_{ys} \rightarrow$ stress state is inside yield surface

$\sigma_{ef}(\{\sigma\}) = \sigma_{ys} \rightarrow$ stress state is on yield surface

$\sigma_{ef}(\{\sigma\}) > \sigma_{ys} \rightarrow$ stress state is outside yield surface

The first step is to calculate the final stress state $\{\sigma_1\}$ assuming no additional yielding (block 1). Block numbers are indicated at the upper left-hand corner of the blocks. If $\sigma_{ef}(\{\sigma_1\}) < \sigma_{ys}$ then $\{\sigma_1\}$ is the correct stress state (block 3A). If not, then $\{\sigma_o\}$ relative to the yield surface is examined (block 3B). If $\sigma_{ys} = \sigma_{ef}(\{\sigma_o\})$, block 4B is followed. If $\sigma_{ys} > \sigma_{ef}(\{\sigma_o\})$, the initial stress state is inside the yield surface. Hence, the strain increment must be divided into two parts: that required to reach the yield surface, $\Delta\bar{\varepsilon}$, and the remainder, $\Delta\hat{\varepsilon}$, which is the strain increment after reaching the yield surface. These strain increments are calculated by solving the equations in block 4A. Next the incremental elasto-plastic matrix $[D^*]$ is calculated. The final stress state is obtained by adding the linear and nonlinear stress increments, $[D]\ \{\Delta\bar{\varepsilon}\}$ and $[D^*]\ \{\Delta\hat{\varepsilon}\}$, respectively (block 6). Note that if $\{\sigma_o\}$ had been on the yield surface, $\{\Delta\bar{\varepsilon}\} = 0$ and $\{\Delta\hat{\varepsilon}\} = \Delta\varepsilon$. Next the yield stress σ_{ys} is updated for strain-hardening materials. Finally, $\{\sigma_1\}$ is scaled back to the new yield surface (block 8).

Input Data

The required input data is described in this section. Where applicable, the maximum allowable values of the input parameters are noted.

Card set	Parameters	No. of cards	Format
1.	TITLE(I), I = 1,60	3	20A4
	TITLE = TITLE OF PROBLEM		
2.	OUTPUT, ANALYS, PLANE, QUADRAT, ENERGY	1	5A8

OUTPUT = Output option

= XLONG for long output

= SHORT for output (the nodal coordinates, element connectivity, and boundary conditions are not in the output)

ANALYS = Type of analysis

= XLINEAR for linear analysis

= GNONLIN for geometrically nonlinear analysis

= PNONLIN for materially nonlinear analysis

= CNONLIN for combined geometric and material nonlinear analysis

PLANE = Option for plane stress/plane strain analysis

= PSTRESS for plane stress

= PSTRAIN for plane strain

QUADRAT = Integration option

= REDUC for reduced integration

= XFULL for full integration

ENERGY = Option for strain-energy release rate calculations

= DOG for G calculation

= DONOJG for no G calculation

Card set	Parameters	No. of cards	Format
3.	ITSTEP, NCYCLE, IMAX	1	3I5

ITSTEP = Number of steps in the incremental loading
minimum = 1, maximum = 30

NCYCLE = Number of iterations between updates of stiffness matrix

IMAX = Maximum number of iterations allowed before terminating

4.	ACCURACY	1	F10.3

ACCURACY = Maximum residual allowed in converged solution

5. NN, NE, NRN

NN = Number of nodes in the FE model, max. = 650

NE = Number of elements in the FE model

NRN = Number of nodes with a restrained degree of freedom

6. Nodal Coordinates:

x-coordinate

XX, N(1) = 1,13	*	E10.4, 13I5

XX = coordinate

N( ) = list of nodes with coordinate XX

*Input until all x-coordinates are specified. End x-coordinate data with a blank card.

y-coordinate

XX, N(I), I = 1,13	*	E10.4, 13I5

*Similar to input of x-coordinates

7.	I, IN(I), JN(I), KN(I), LN(I)	NE	5I5

I,IN,JN,KN,LN = Element number, four node numbers for element I. Nodes must be specified in a counterclockwise direction.

Card set	Parameters	No. of cards	Format
8.	K, NRL (2*K-1), NRL (2*K)	NRN	3I5

K = Node number

NRL (2*K-1), NRL (2*K) = Constraints in X and Y directions, respectively, at node K.
0 indicates no constraint
1 indicates constraint

Note: Do not include degrees of freedom involved in multipoint constraints. Do include degrees of freedom with specified displacements.

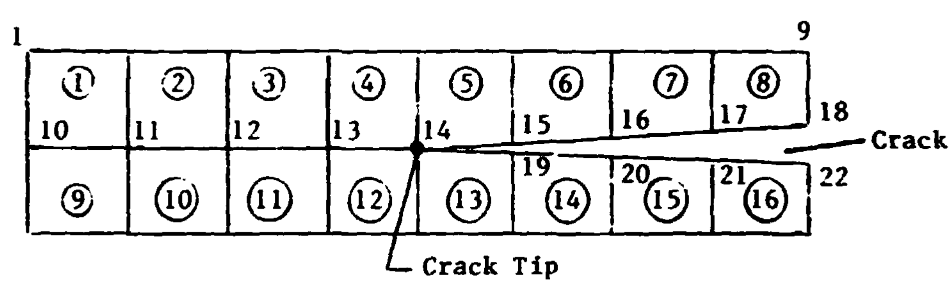
SKIP 9-12 IF ENERGY = DONCJG

Card set	Parameters	No. of cards	Format
9.	INP	1	I5

INP = Number of node sets used in virtual crack extension calculation (maximum = 15)

Card set	Parameters	No. of cards	Format
10.	NEGCAL(I), I = 1, (INP+1)	(INP+1)/16†	16I5

NEGCAL = Element numbers for elements contributing to the nodal forces required for virtual crack extension. (See example in sketch below. Element numbers are circled.)



IF INP = 3,
NEGCAL (1 to 4) = 2, 3, 4, 5
NFGCAL (1 to 3) = 14, 13, 12
NDGCAL (1 to 6) = 15, 19, 16, 20, 17, 21

†Round off to next higher integer.

Card set	Parameters	No. of cards	Format
11.	NFGCAL(I), I = 1, INP	INP/16[†]	16I5

NFGCAL(I) = Node numbers for nodes along which virtual crack extension forces are calculated. List according to distance from crack tip, with the crack tip node as the first one. (See sketch above.)

12.	NDGCAL(I), I = 1, (2*INP)	2*INP/16[†]	16I5

NDGCAL(I) = Node numbers for the nodes used to calculate cracking opening and sliding displacements

Repeat card sets 13-16 for each material group.

Maximum number of material groups = 10

End last group with blank card.

13.	J, XMATER(J)	1	I5, A8

J = Material group number

XMATER = Material type

= ELASTIC for linear stress-strain curve

= ELPLAST for elastic-perfectly plastic stress-strain curve

= BLINEAR for bilinear stress-strain curve

= RAMOSGO for Ramberg-Osgood stress-strain curve

14.	EX, EY, PYX, GXY	1	4E10.3

E_x, E_y, ν_{yx}, G_{xy}:

E_x Young's modulus in x-direction

E_y Young's modulus in y-direction

$\nu_{yx} = -\frac{\epsilon_x}{\epsilon_y}$ = Poisson's ratio

= Contraction in x-direction due to unit applied strain in y-direction

G_{xy} Shear modulus

[†]Round off to next highest integer.

ORIGINAL PAGE IS
OF POOR QUALITY

Card set	Parameters	No. of cards	Format
15.	YIELDS, ET, RO, ANM	1	5E10.3

YIELDS = Yield stress

ET = Tangent modulus for yielded bilinear material

RO, ANM = Parameters defining Ramberg-Osgood stress-strain relation, $\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{RO}\right)^{ANM}$

(a) If XMATER = ELASTIC, input YIELDS = ET = RO = 1.0×10^{21}, ANM = 10

(b) If XMATER = BLINEAR, input proper YIELDS and ET and set RO = ANM = 0.0

(c) If XMATER = RAMOSGO, input proper YIELDS, RO and ANM and set ET = 0.0

Card set	Parameters	No. of cards	Format
16.	NEL1, NEL2, NELINC	*	3I5

NEL1, NEL2, NELINC = Loop parameters used to define elements in material group

NEL1 = First element

NEL2 = Last element

NELINC = Loop increment

e.g., 1, 50, 20 defines elements 1, 21, and 31 to be in material group

*Repeat until all elements in group are defined.
End card set 16 by specifying NEL1 = NEL2 = NELINC = 0

Card set	Parameters	No. of cards	Format
17.	DELLOAD(I) = 1, ITSTEP	ITSTEP/8†	8F10.3

DELLOAD(I) = Scale factor for proportional load vector for load step I. Always specify DELLOAD(1) = 1.0

†Round off to next higher integer.

Card set	Parameters	No. of cards	Format
18.	NLN, NCD, NED	1	3I5

NLN = Number of nodes with applied loads

NCD = Number of multipoint constraints, max = 15

NED = Number of specified displacements, max = 30

19.	K, FX, FY	NLN	I5, 2F10.3

K = Node number

FX,FY = Loads in x and y directions, respectively

20.	K, KDF, URD	NED	2I5, F10.3

K = Node number

KDF = Displacement direction, specify 1 for x direction
specify 2 for y direction

URD = Magnitude of displacement

SKIP 21-24 IF NCD = 0

21.	NMPR(I), I = 1, NCD	NCD/16	16I5

NMPR(I) = Number of degrees of freedom involved in the Ith multipoint constraint. max = 20

22.	((ICDN(I,J),J=1,NMPR(I)), I = 1, NCD)	NCD sets	16I5

ICDN(I,J) = Jth degree of freedom involved in the Ith multipoint constraint

Start a new card for each multipoint constraint.
Start with lowest number degree of freedom.

23.	NZKV	1	I5

NZKV = Number of multipoint constraints for which there is an applied load

24.	NKV, ATOT	NZKV	I5, F10.3

NKV, ATOT: ATOT is the non-zero load associated with the NKV set of constrained nodes

References

1. Dattaguru, L.; Everett, R. A.; Whitcomb, J. D.; and Johnson, W. S.: Geometrically Nonlinear Analysis of Adhesively Bonded Joints. NASA TM 84562, Sept. 1982.

2. Mall, S.; Johnson, W. S.; and Everett, R. A., Jr.: Cyclic Debonding of Adhesively Bonded Composites. NASA TMX 84577, Nov. 1982.

3. Everett, R. A., Jr.: The Role of Peel Stresses in Cyclic Debonding. Adhesive Age, Vol. 26, No. 5, May 1983.

4. Zienkiewicz, O. C.: The Finite Element Method in Engineering Science, 2nd Ed., McGraw-Hill, New York, 1971.

5. Owen, D. R. J.; and Hinton, E.: Finite Elements in Plasticity: Theory and Practice. Pineridge Press Limited, Swansea. U. K., 1980.

6. Rybicki, E. F.; and Kanninen, M. F.: A Finite Element Calculation of Stress Intensity Factors by a Modified Crack Closure Integral, Engineering Fracture Mechanics, 1977, Vol. 9, pp. 931-938.

7. Pawsey, S. F.; and Clough, R. W.: Improved Numerical Integration of Thick Shell Finite Elements, in International Journal for Numerical Methods in Engineering, Vol. 3, 1971, pp. 575-586.

8. Zienkiewicz, O. C.; Taylor, R. L.; and Too, J. M.: Reduced Integration Technique in General Analysis of Plates and Shells, in International Journal for Numerical Methods in Engineering, Vol. 3, 1971, pp. 275-290.

9. Whitcomb, J. D.: Finite Element Analysis of Instability-Related Delamination Growth. Journal of Composite Materials, Vol. 15, Sept. 1981, pp. 403-426.

10. Cook, R. D.: Concepts and Applications of Finite Element Analysis. John Wiley & Sons, Inc.

ORIGINAL PAGE IS
OF POOR QUALITY

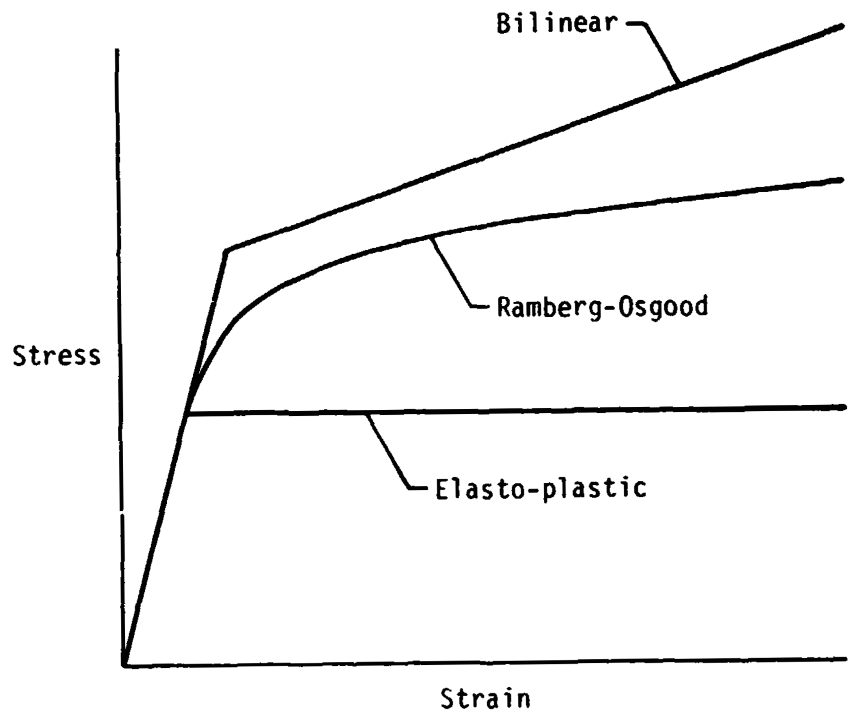
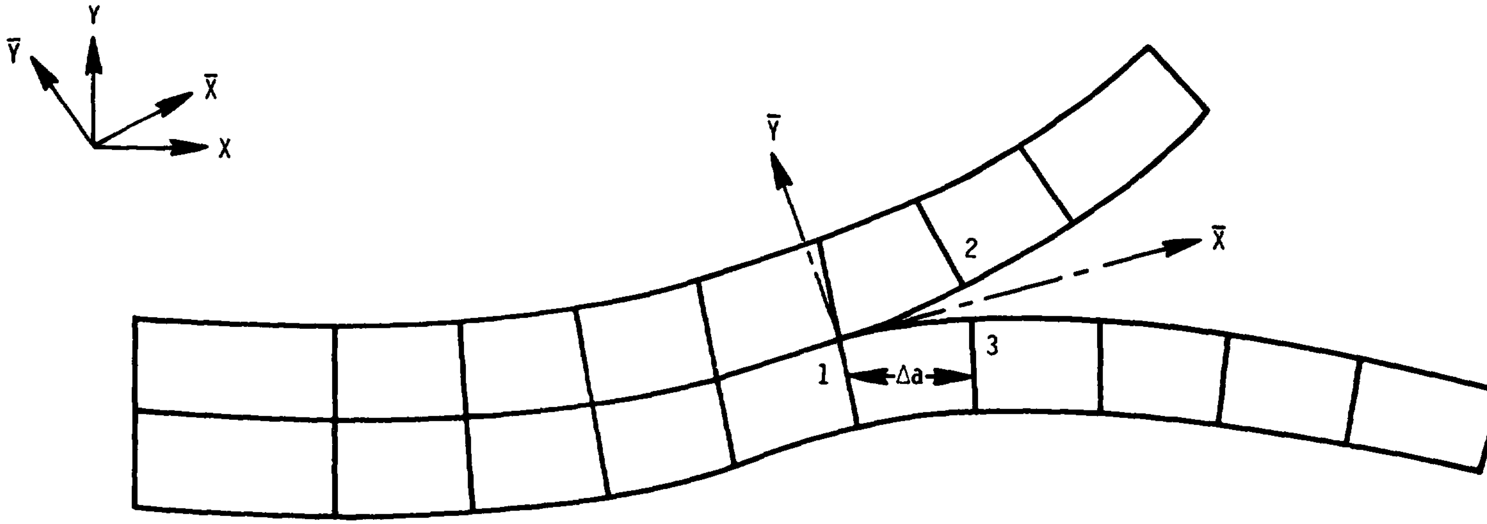
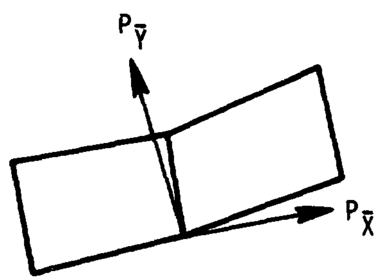


Figure 1.- Types of uniaxial stress-strain curves.





$$\mathcal{G}_{I} = 1/2\, P_{\bar{Y}} \frac{\left(\bar{v}_2 - \bar{v}_3\right)}{\Delta a}$$

$$\mathcal{G}_{II} = 1/2\, P_{\bar{X}} \frac{\left(\bar{u}_2 - \bar{u}_3\right)}{\Delta a}$$

Figure 2.- Transformed coordinate system for strain-energy-release rate calculation.

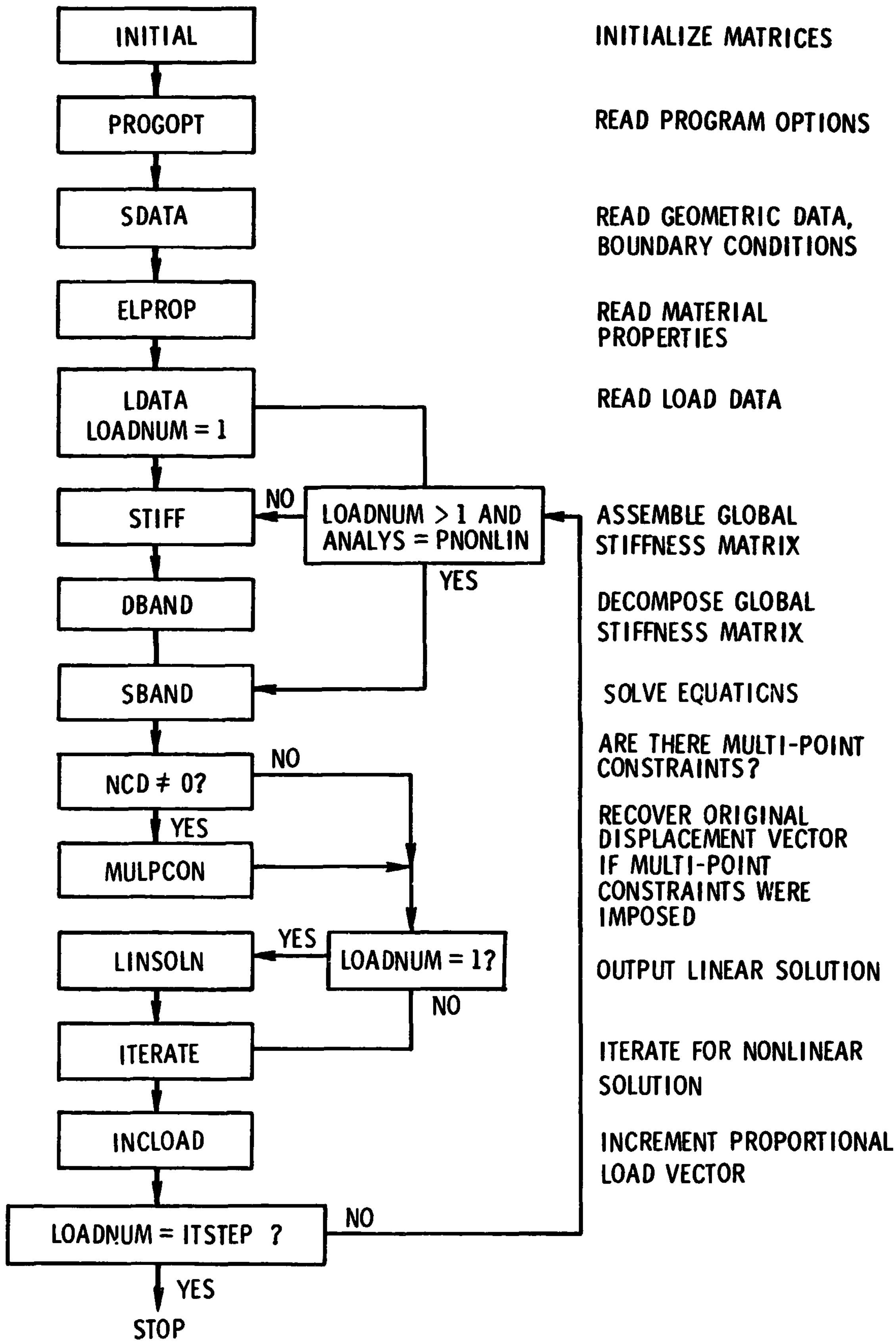


Figure 3.- Flow chart for main program.

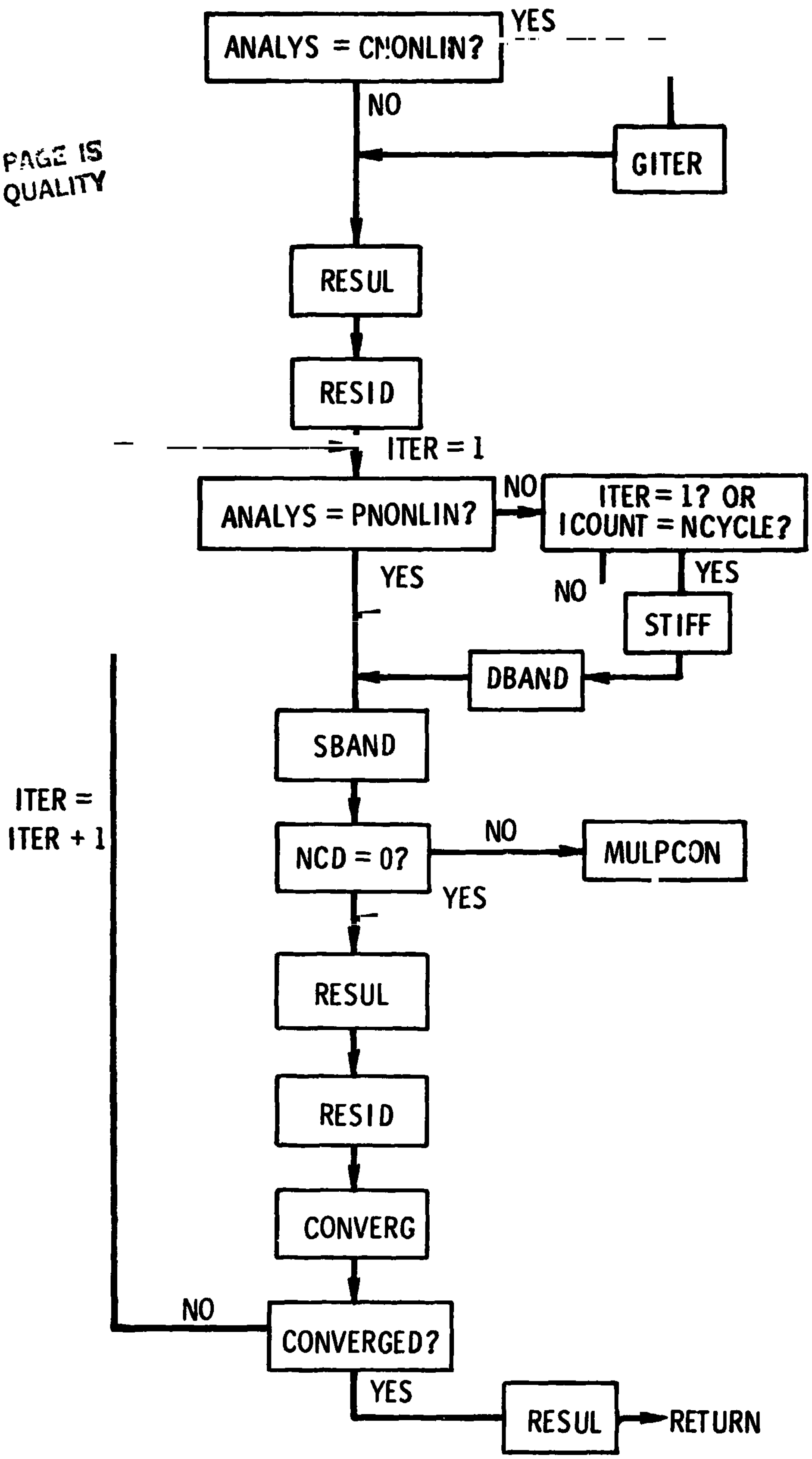


Figure 4.- Flow chart for subroutine Iterate.

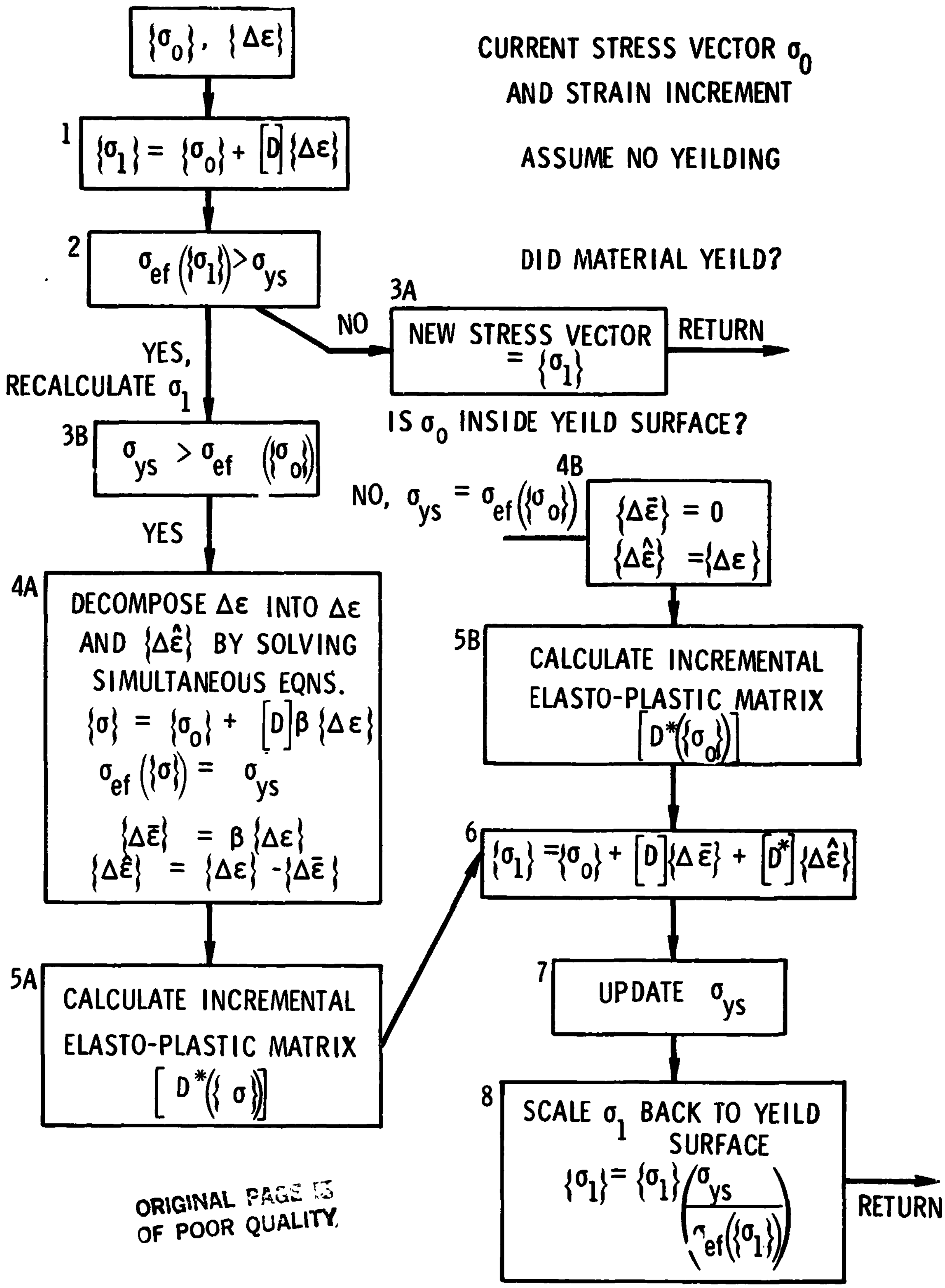


Figure 5.- Flow chart for subroutine STRSCAL.

APPENDIX A

This appendix gives the names and function of the subroutines and the major program variables.

Subroutines

NAME	FUNCTION
1. BLKSIGM	Calculates submatrices for element initial stress matrix
2. BLMAT	Calculates nonlinear component of strain-displacement matrix
3. BMAXQ4	Calculates linear component of strain-displacement matrix
4. CFILL	Fills matrix of element nodal coordinates
5. CONVERG	Checks for convergence
6. DATA	Reads nodal coordinate data
7. DBAND	Performs Cholesky decomposition on global stiffness matrix
8. DEPMAT	Calculates elastic-plastic matrix, $[D^*_{ep}]$
9. ELPROP	Reads material properties
10. FORCEP	Calculates internally generated nodal forces for an element
11. GCAL	Calculates strain-energy release rates
12. GITER	Solves nonlinear equations
13. IDVEC	Fills vector of element degrees of freedom
14. INCLOAD	Scales load vector
15. INITIAL	Initializes variables
16. ITERATE	Solves nonlinear equations
17. KLARGE	Calculates element large deflection stiffness matrix
18. KSIGNEW	Calculates element initial stress matrix
19. LDATA	Reads load data
20. LINSOLN	Outputs linear solution
21. MATMUL	Performs matrix multiplication

22. MULPCON Modifies stiffness matrix and displacement vector for multi-point constraints

23. PROGOPT Reads program options

24. RCADD Adds rows and columns of the stiffness matrix

25. RESID Calculates residual force vector

26. RESUL Calculates strains and stresses

27. SBAND Solves set of linear equations. (Used with DBAND)

28. SDATA Reads structural data

29. STAXQ4 Calculates linear element stiffness matrix

30. STIFF Assembles global stiffness matrix

31. STRSCAL Calculates incremental stresses from incremental strain

32. TRANS Generates transpose of a matrix

Program Variables (Arrays are shown with their dimensions.)

Variable	Definition
AN (1300)	Incremental load vector
ANALYS	Type of analysis
ANM	Exponent in Ramberg-Osgood equation for uniaxial stress-strain curve $\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{RO}\right)^{ANM}$
ANTOTAL (1300)	Total load vector
AR (1300)	Nodal restraint force vector
ATOT	Load associated with multipoint constraint
DELLOAD (30)	Scale factor for incremental loads
DISP (1300)	Incremental displacement vector
DN (1300)	Total displacement vector
DPS (30)	Vector of specified non-zero displacements
ENERGY	Option for strain-energy release rate calculation

FF (1300,4)	Effective stresses at the end of an increment
FI (1300,4)	Effective stresses at the beginning of an increment
FXX (10)	X-direction forces used in strain-energy-release rate calculation
FYY (10)	Y-direction forces used in strain-energy-release rate calculation
IBW	Bandwidth of global stiffness matrix
ICDN (20,15)	Degrees of freedom involved in multi-point constraints
IN (1300), JN (1300), KN (1300), LN (1300)	Element connectivity arrays. Connectivity for element number I is IN(I), JN(I), KN(I), LN(I)
INP	Number of node sets used in virtual crack closure calculation of strain-energy release rates
IPE (1300)	List of yielded elements (only used for output)
ITSTEP	Number of incremental load steps
LOADNUM	Incremental load step number
MATER (1300)	Element material group numbers
MIBW	Number of columns in global stiffness matrix, SN. Currently MIBW = 70
MRANK	Number of rows in global stiffness matrix, SN. Currently MRANK = 1300
NCD	Number of multipoint constraints
NDPS (30)	Vector of degrees of freedom with specified non-zero values
NE	Number of elements in finite element model
NED	Number of specified displacements
NLN	Number of nodes with applied forces
NMPR	Number of degrees of freedom involved in a set of multipoint constraints
NN	Number of nodes in finite element model

NND	Number of degrees of freedom in finite element model before applying boundary conditions
NRL (1300)	Degree of freedom restraint list
NRN	Number of nodes with a restrained degree of freedom
OUTPUT	Output option
PLANE	Plane stress/plane strain option
PSI (1300)	Residual force vector
QUADRAT	Integration option
RO	Parameter in Ramberg-Osgood equation. See definition for "ANM"
SGYBAR (1300,4)	Current yield stress
SN (1300,70)	Global stiffness matrix
STRESS (1300,12)	Stresses
T3 (10,3,3)	Elasticity matrices for the material groups
UX (10)	Tangential displacements vector for strain-energy-release rate calculation
UY (10)	Opening displacements vector for strain-energy-release rate calculation
X (1300)	Nodal x-coordinates
Y (1300)	Nodal y-coordinates

APPENDIX B

This appendix gives input data and results for three samples problems.

The first problem (fig. B-1a) involves transverse displacement of a long thin rod. The finite element mesh is shown with node and element numbers and boundary conditions. The left end is pinned; the right end can move only in the "y" direction. The transverse displacement, v, at node 9 was specified because the initial transverse stiffness is zero, which would have caused a singular stiffness matrix if a transverse load had been specified. Although the rod initially has zero transverse stiffness, geometrically nonlinear effects stiffen the system as the transverse displacement increases. Figure B-1b shows the calculated axial stress in the rod (element 2) as a function of lateral displacement. The finite element results are shown as symbols. The two curves are exact solutions, derived using simple trigonometry, for a rod under axial load. One curve is for a linear elastic material and the other is for a elasto-perfectly plastic material with a yield stress of 50 KSI. The finite element analysis predicts the nonlinear response very well. The differences between the exact results and the finite element results are due to the very coarse mesh and the end restraints not being along the rod's longitudinal axis. Table B-1 lists the numerical values at the element centroids calculated by GAMNAS.

Figure B-2 shows the input data for the linear elastic rod. Required changes to this data for the elasto-perfectly plastic rod are shown in parenthesis.

The second problem involves transverse loading of a double cantilever beam. Figure B-3 shows the finite element model, which has 50 nodes and 32 elements. Two versions of the finite element analysis were used: one version used full integration and one used reduced integration. The input data for analysis with reduced integration are shown ir g. B-4. The change required for full integration is shown in parenthesis.

The strain energy release rate (using strength of materials) is given by

$$G = \frac{M^2}{EI} \tag{B1}$$

A transverse load of 20 lb. was used, resulting in a moment of 40 in./lb. From eqn. (B1), G is calculated to be 1.92 lb/in. The full and reduced integration yielded 1.45 lb/in. and 1.97 lb/in., respectively. Even with a coarse mesh, the reduced integration version yielded an accurate result. The full integration version illustrates the well-known poor performance of isoparametric quadrilaterals in modeling bending deformation.

The final problem (see fig. B-5a) involves polar symmetric loading of a rectangular region. By imposing appropriate boundary conditions along $x = 0$, only half of the region needed to be modeled. The polar symmetric conditions are imposed using multi-point constraints to specify $u(o,y) = -u(o,-y)$ and $v(o,y) = -v(o,-y)$.

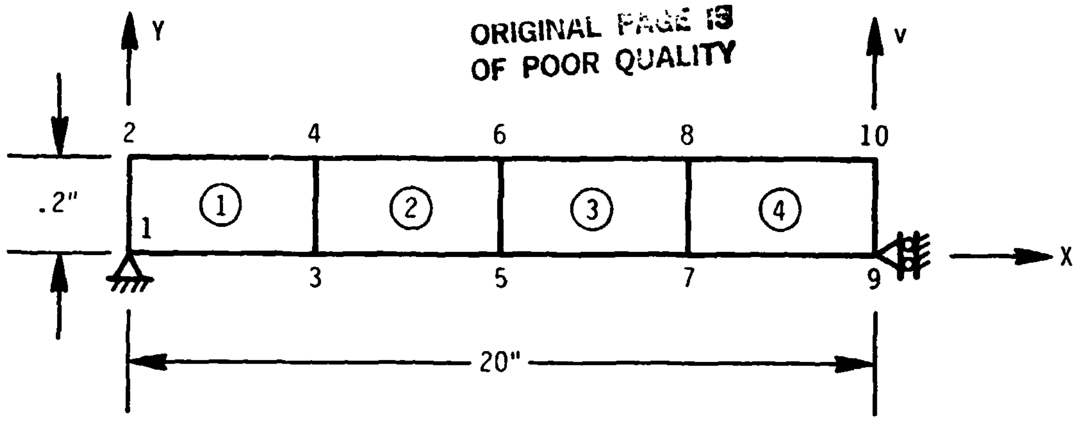
Figure B-5 shows the finite element model before and after loading. Table B-2 gives the numerical values of the nodal displacements. The required input data are shown in fig. B-6.

TABLE B-1 AXIAL STRESS IN LONG THIN ROD

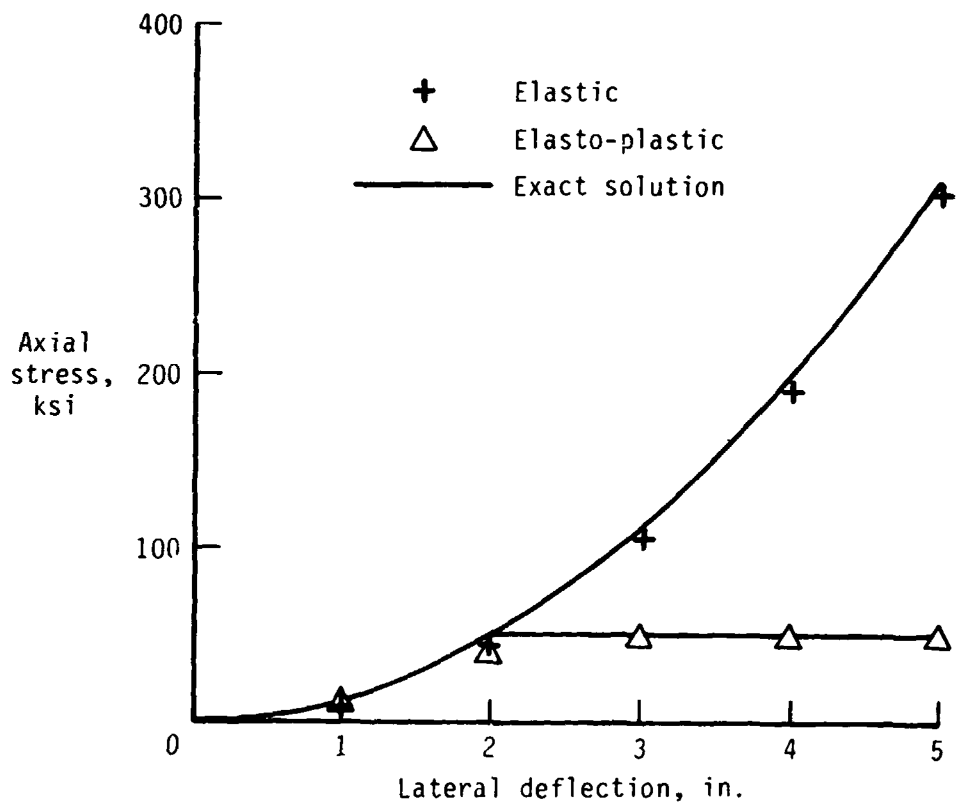
LATERAL DEFLECTION INCHES	AXIAL STRESS, KSI ELEMENT 1	2	3	4	MATERIAL TYPE
1	8.868	8.874	8.874	8.868	LINEAR
2	43.48	43.52	43.52	43.48	
3	104.8	104.9	104.9	104.8	
4	191.7	191.9	191.9	191.7	
5	303.7	303.9	303.9	303.7	
1	8.868	8.874	8.874	8.868	NONLINEAR
2	40.03	40.09	40.09	40.03	
3	48.61	48.68	48.68	48.61	
4	49.53	49.60	49.60	49.53	
5	49.84	49.91	49.91	49.84	

TABLE B-2 NODAL DISPLACEMENTS (INCHES) FOR RECTANGULAR REGION WITH POLAR SYMMETRIC LOADS

NODE	$u \times 10^4$	$v \times 10^4$	NODE	$u \times 10^4$	$v \times 10^4$
1	-1.095	.08053	14	1.291	-.4773
2	-.6526	.02078	15	2.146	-.6216
3	$.2 \times 10^{-23}$	$.2 \times 10^{-23}$	16	-.4933	-1.063
4	.6526	-.02078	17	-.4438	-1.028
5	1.095	-.08053	18	.2080	-1.140
6	-.9096	-.2388	19	1.574	-1.100
7	-.4577	-.2521	20	3.415	-1.268
8	.1880	-.2250	21	-.1146	$-.5 \times 10^{-23}$
9	.9405	-.2243	22	-.3693	-1.195
10	1.462	-.3232	23	.3345	-2.047
11	-.7632	-.6573	24	1.532	-2.900
12	-.3817	-.6648	25	5.721	-3.731
13	.2632	-.5427			



a) FINITE ELEMENT MODEL FOR A LONG THIN ROD



b) AXIAL STRESS VS. SPECIFIED TRANSVERSE DISPLACEMENT

Figure B-1.- Transverse displacement of a long thin rod.

ORIGINAL PAGE IS
OF POOR QUALITY

```
LONG SLENDER BEAM-COLUMN                    COSMIC-1
******************************************************
******************************************************
SHORT   GNONLIN PSTRESS REDUC   DONDJG          (CHANGE GNONLIN TO CNONLIN)
    5    2   20
     0 010
   10    4    2
0 0           1    2
5 0           3    4
10            5    6
15            7    8
20            9   10
0000000000000000000000000000
0 0           1    3    5    7    9
 2            2    4    6    8   10
0000000000000000000000000000
    1    1    3    4    2
    2    3    5    6    4
    3    5    7    8    6
    4    7    9   10    8
    1    1    1
    9    1    1
    1ELASTIC                                    (CHANGE ELASTIC TO ELPLAST)
 0 100E+08 0 100E+08 0 300E+00 0 385E+07
 0 500E+05 0 000E+00 0 000E+00 0 000E+00
    1    4    1
    0    0    0

     1 000     2 000     3 000     4 000     5 000
    0    0    1
    9    2     1 000
```

Figure B-2.- Input file for linear elastic rod. Changes required for elasto-plastic rod are shown in parentheses.

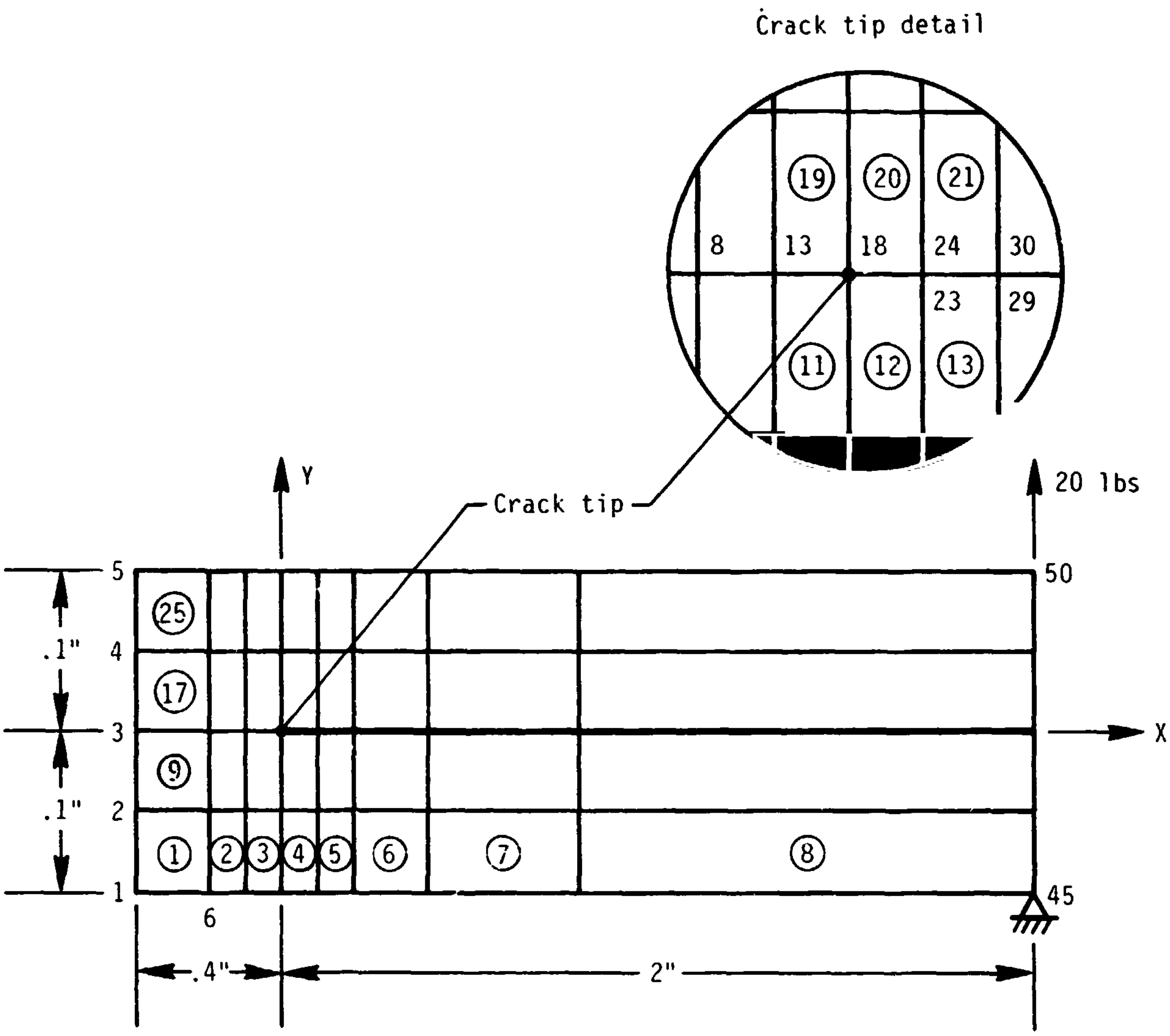
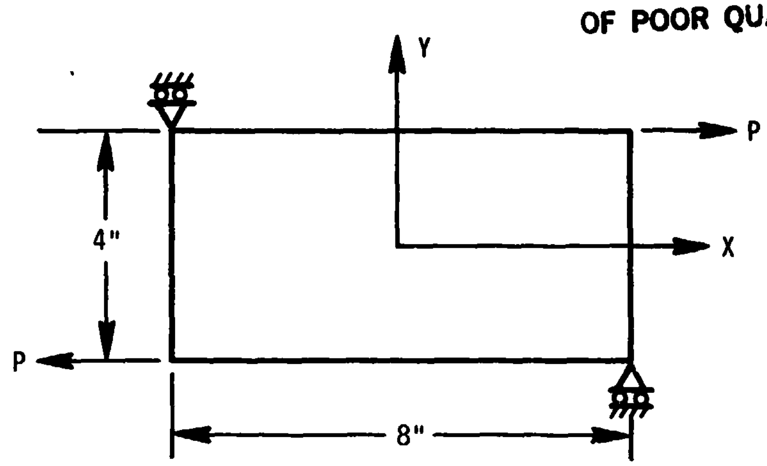


Figure B-3.- Finite-element model for double-cantilever beam. Crack extends from X = 0.0 to 2.0 along the line Y = 0.0.

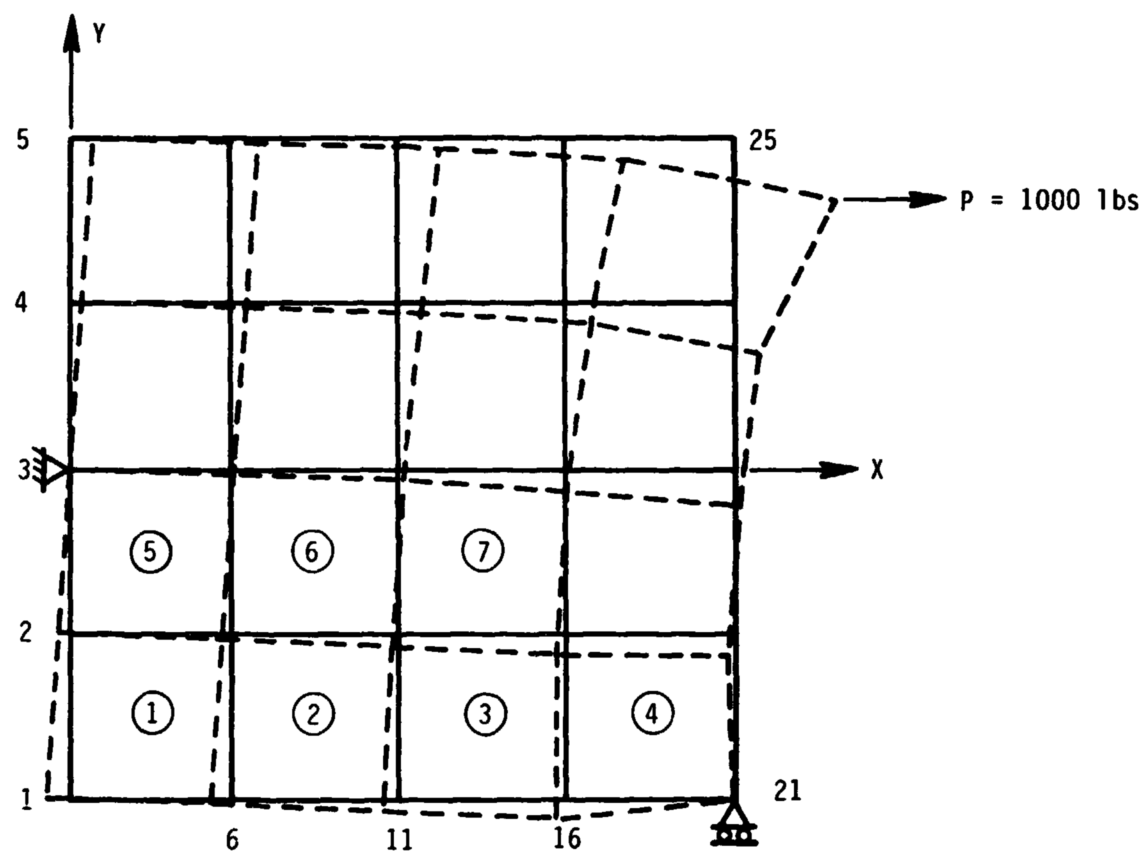
ORIGINAL PAGE IS
OF POOR QUALITY

```
DOUBLE CANTILEVERED BEAM                    COSMIC-3
********************************************************************
********************************************************************
SHORT   XLINEAR PSTRESS REDUC   DOG                    (CHANGE REDUC TO XFULL)
    1     1     1
    1 000
   50    32     2
- 4000E+00      1     2     3     4     5
- 2000E+00      6     7     8     9    10
- 1000E+00     11    12    13    14    15
0 0000E+00     16    17    18    19    20
0 1000E+00     21    22    23    24    25    26
0 2000E+00     27    28    29    30    31    32
0 4000E+00     33    34    35    36    37    38
0 8000E+00     39    40    41    42    43    44
0 2000E+01     45    46    47    48    49    50

- 1000E+00      1     6    11    16    21    27    33    39    45
- 5000E-01      2     7    12    17    22    28    34    40    46
0 0000E+00      3     8    13    18    23    29    35    ′1    47
0 0000E+00     24    30    36    42    48
0 5000E-01      4     9    14    19    25    31    37    43    49
0 1000E+00      5    10    15    20    26    32    38    44    50

    1     1     6     7     2
    2     6    11    12     7
    3    11    16    17    12
    4    16    21    22    17
    5    21    27    28    22
    6    27    33    34    28
    7    33    39    40    34
    8    39    45    46    40
    9     2     7     8     3
   10     7    12    13     8
   11    12    17    18    13
   12    17    22    23    18
   13    22    28    29    23
   14    28    34    35    29
   15    34    40    41    35
   16    40    46    47    41
   17     3     8     9     4
   18     8    13    14     9
   19    13    18    19    14
   20    18    24    25    19
   21    24    30    31    25
   22    30    36    37    31
   23    36    42    43    37
   24    42    48    49    43
   25     4     9    10     5
   26     9    14    15    10
   27    14    19    20    15
   28    19    25    26    20
   29    25    31    32    26
   30    31    37    38    32
   31    37    43    44    38
   32    43    49    50    44
   45     1     1
   50     1     0
    1
   19    20
   18
   23    24
    1ELASTIC
 0 100E+08 0 100E+08 0 300E+00 0 385E+07
 0 000E+00 0 000E+00 0 000E+00 0 000E+00
    1    32     1
    0     0     0

     1 000
    1     0     0
   50     0 000     20 000
```

Figure B-4.- Input file for reduced integration analysis of double-cantilever beam. The change required for full integration is shown in parenthesis.



a) RECTANGULAR REGION WITH POLAR SYMMETRIC LOADS



b) ORIGINAL AND DEFORMED CONFIGURATIONS

Figure B-5.- Polar symmetric loading of a rectangular region.

```
POLAR-SYMMETRIC LOADING OF RECTANGULAR REGION
********************************************    FILE=COSMIC-5
************************************************************************
XLONG    XLINEAR PSTRESS XFULL    DONOJG
    1     1     1
     1 000
   25    16     2
0 0000E+00     1     2     3     4     5
0 1000E+01     6     7     8     9    10
0 2000E+01    11    12    13    14    15
0 3000E+01    16    17    18    19    20
0 4000E+01    21    22    23    24    25

0 0000E+00     1     6    11    16    21
0 1000E+01     2     7    12    17    22
0 2000E+01     3     8    13    18    23
0 3000E+01     4     9    14    19    24
0 4000E+01     5    10    15    20    25

    1     1     6     7     2
    2     6    11    12     7
    3    11    16    17    12
    4    16    21    22    17
    5     2     7     8     3
    6     7    12    13     8
    7    12    17    18    13
    8    17    22    23    18
    9     3     8     9     4
   10     8    13    14     9
   11    13    18    19    14
   12    18    23    24    19
   13     4     9    10     5
   14     9    14    15    10
   15    14    19    20    15
   16    19    24    25    20
    3     1     1
   21     0     1
    1ELASTIC
 0 100E+08 0 100E+08 0 300E+00 0 385E+07
 0 000E+00 0 000E+00 0 000E+00 0 000E+00
    1    16     1
    0     0     0

     1 000
    1     4     0
   25  1000 000      0 000
    2     2     2     2
   -1    -9
   -3    -7
   -2   -10
   -4    -8
    0
```

Figure B-6.- Input file for polar symmetric loading of rectangular region.

APPENDIX C

This appendix discusses error messages and potential debug strategies.

A self-explanatory diagnostic message is output and execution terminated under the following conditions:

1) A node has an unspecified coordinate

2) An element has an unspecified material group number

3) The plane stress or plane strain option is spelled incorrectly

4) An element has a linear stiffness matrix with a diagonal element less than or equal to zero.

5) The rank or bandwidth of the global stiffness matrix exceeds the maximum allowed.

If the global stiffness matrix is singular, the decomposition routine, DBAND, prints "Matrix is singular" and halts execution. Failure to specify sufficient restraints to prevent rigid body motion is a frequent cause for a singular stiffness matrix. A singular stiffness matrix is often encountered in geometrically nonlinear analysis because the load increments are too large (which causes the iterative solution process to diverge) or because buckling occurs. The maximum allowable load increment can only be determined through experience. However, frequent updating of the tangential stiffness matrix (i.e., a small value is input for NCYCLE) does permit larger load increments.

The internally generated forces at all nodes are calculated and output. These forces should be numerically zero except at nodes where loads are applied or displacements are specified, or at nodes involved in a multi-point constraint. Errors in modeling will often cause spurious nodal forces, which can be used to help isolate the modeling errors.

Plotting all finite element models is highly recommended, since the plot will quickly reveal many input errors. To track down errors not diagnosed by GAMNAS, host computer debug utilities are recommended.